ISSN (O): 2393-8609

International Journal of Aerospace and Mechanical Engineering
Volume 2 – No.5, September 2015

Design and Analysis of Wing Braces for Amphibious Aircraft

Priyanshu Srivastava
Student
B.tech(Aerospace)
Gurukul Vidyapeeth
priyanshu.srivastava92@gmail.com

Deepak kumar
Student
B.tech(Aeronautical)
Gurukul Vidyapeeth
deepakgujjar45@gmail.com

Loveleen
Student
B.tech(Aeronautical)
Gurukul Vidyapeeth
loveleenk62@gmail.com

Mohar Singh
Student
B.tech(Aerospace)
Gurukul Vidyapeeth
loveleenk62@gmail.com

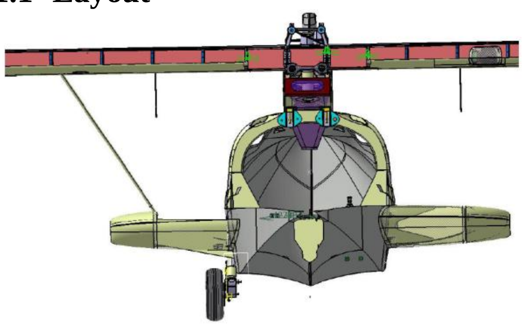
Neeraj
Student
B.tech(Aeronautical)
Gurukul Vidyapeeth
loveleenk62@gmail.com

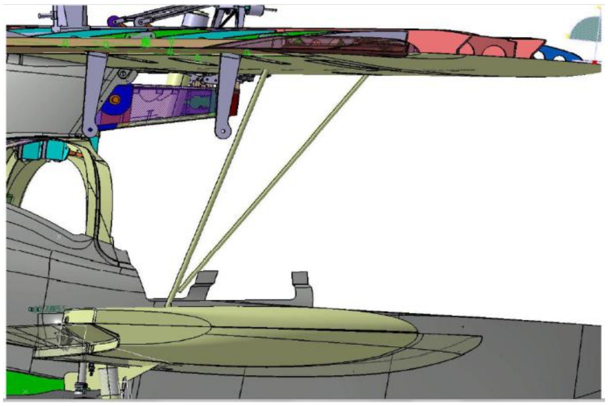
ABSTRACT
The paper is about the Design and FE analysis of Wing braces of an amphibious aircraft. Braces maintain the shape of wing and also support the bending and compressive load which act on the wing during landing and takeoff. Braces are attached to the spars and same rib via a bracket. It act as a load distributor. CATIA V5 software is used for modeling. The stress analysis of braces is carried out to compute the stresses at bace structure. FE analysis is used to find out the saftey factor of the structure. It shows the maximum stress location. Linear static and buckling analysis has been analyzed in ANSYS.

Keywords
FE Analysis, Braces, Rib, Bracket, Linear static and Buckling load.

1. INTRODUCTION
For some optical and structural reasons the wing should be supported with two braces per side to the floats. The rear brace is connected via a bracket to the rear wing spar and to a rib along x axis. The front brace is connected via a bracket to the front spar and the same rib. The brackets include a hinge below the lower wing surface, where the brace is connected. Both braces are brought together at one single lower point, which is connected to the float with a quick release pin. After releasing the pin the braces should be able to be turned in the upper hinges and fixed on the lower wing surface.

1.1 Layout





1.2 Market Survey
A market survey is carried out to find the diameters or radius of connecting rod available in the market. This is done to cut down the cost of manufacturing. by using rods available in the market saves both time and cost of manufacturing. A rod of suitable diameter is selected.

1.3 CATIA V5
CATIA (Computer Aided Three Dimensional Interactive Application) is a multi platform CAD/CAM/CAE commercial software suit developed by French company Dassault Systems. It supports multiple stages of product development, including conceptualization, design, engineering and manufacturing. CATIA offers a solution to shape design, styling, surfacing workflow and visualization to create, modify and validate complex innovative shapes from industrial design to class-a surfacing with ICEM surfacing technologies.

1.4 FE Analysis
Finite element analysis is a type of computer program that uses the finite element method to analyze a material or object and find how applied stresses will affect the material or design. FEA can help determine any point of weakness in a design before it manufactured. The analysis is done by create a mesh of points in shape of object that contains information about the material and the object at each point for analysis. FEA can also analyze the effect of vibrations, fatigue, and heat transfer.

ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 2 – No.5, September 2015

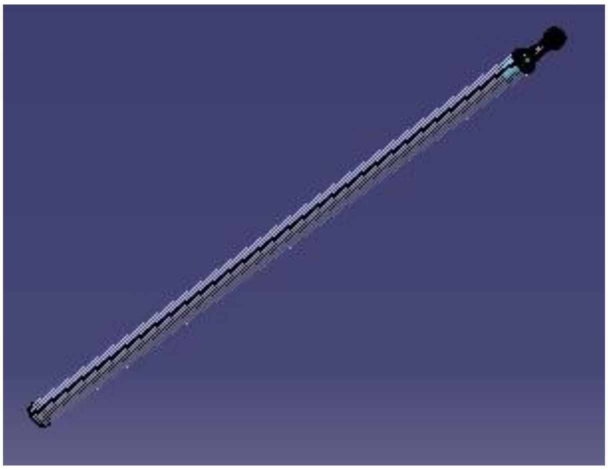
1.5 ANSYS

ANSYS is an engineering simulation software (computer-aided engineering, or CAE). It allows engineers to test systems by simulating fluid flows in virtual environment. It is a finite element analysis software for structural physics that could simulate static(stationary), dynamic(moving), and thermal(heat transfer problems)

2. CATIA MODEL

Brace is modelled in CATIA-V5 has been shown in figure 1

Component	Material	Thickness	Young's Modulus	Poisson ratio
Wing Brace	Al 6061, Stainless steel	1.59	70000 MPa	0.33



9 Fig 1: Brace 3D model

3. MESHING

The model is meshed using tetra element for better results as shown below.

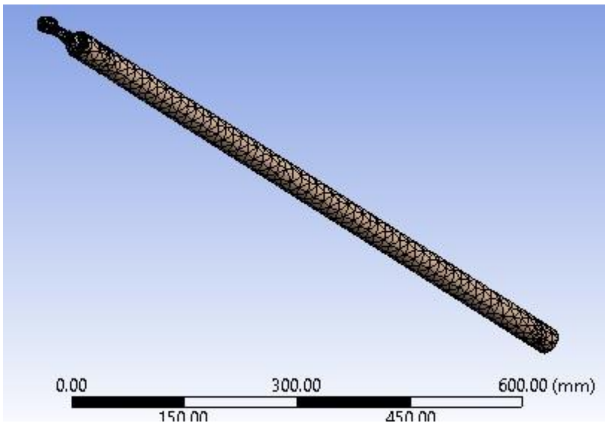


Fig 2: Meshing of brace model

4. LOAD CONDITIONS

Load in front strut along strut axis (N)	Load in rear strut along strut axis (N)	Inertia load factor (g)	Aircraft speed (kn)
24050	18800	3.8	96
14640	24300	3.8	140
-5444	25730	1.9	79
-21300	1587	-1.9	96
-22630	6980	-1.9	79

Note: - positive loads are tension, negative are compression - Positive inertia load factor is acting downward on brace

5. RESULT

The following are the results for load case:- 22630 MPa

5.1 Stress

Stress produced is 225.94 MPa which is far below the allowable yield value of the material.

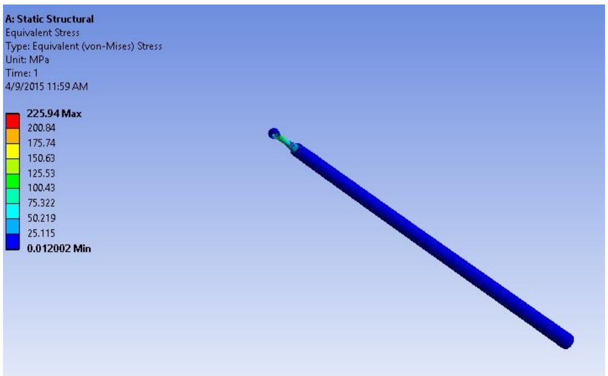


Fig 3: Stress produced in model

5.2 Total deformation

Total deformation produced is 0.082053mm as shown in the figure below.

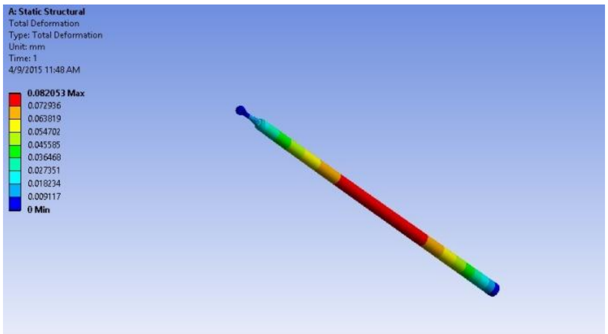


Fig 4: Total deformation produced in model

5.3 Buckling

Buckling is produced as shown in the figure below.

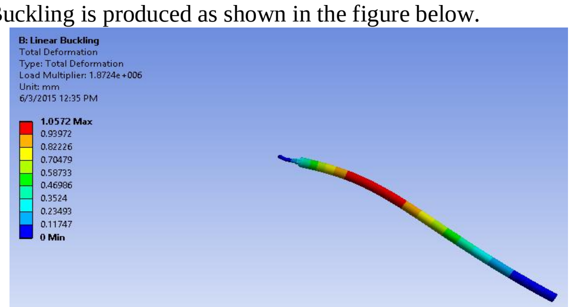


Fig 5: Buckling produced in figure

ISSN (O): 2393-8609

International Journal of Aerospace and Mechanical Engineering

Volume 2 – No.5, September 2015

Critical buckling load is far beyond the applied load. Hence design is safe.

6. CONCLUSION

Design and FE analysis of brace structure is carried out and maximum stress, deformation and buckling is identified which is found out to be lower than the yield strength of material.

7. REFERENCES

[1] Book: Rorak's formula for stess and strain (WARREN C. YOUNG, RICHARD G.BUDYNAS)

[2] Brown, L. D., Hua, H., and Gao, C. 2003. A widget framework for augmented interaction in SCAPE.

[3] CS-VLA